6. DEFINING BOUNDARY CONDITIONS

6.1. Overview of Boundary Conditions

Chapter 5 describes procedures for drawing a cross-section geometry and assigning materials to components, which are the first steps toward performing a heat-transfer analysis. The next step is to define the boundary conditions for the cross section. The procedures for defining boundary conditions are not dependent on how the cross section was drawn.

The steps for assigning boundary conditions are summarized below and explained in detail in the sections that follow.

- Click on the **Boundary Conditions** toolbar button (or use the **Draw/Boundary Conditions** menu choice or the **F10** keyboard shortcut). This causes THERM to draw the boundary conditions for the cross section. Under most circumstances, boundary conditions are drawn automatically around the outside of the completed cross section (see Section 6.5, "Special Boundary Condition Cases" for exceptions). THERM moves around the model counterclockwise and indicates the boundary by a heavy line.

- The program checks for characteristics in the geometry that will cause a problem in the heat-transfer simulation. These include undefined voids, overlapping polygons (see Section 6.3.2, "Finding Voids and Overlaps"), and polygons that are not contiguous. If problems are detected, THERM will issue a message. You may need to resolve these problems before the boundary conditions can be defined. When you have fixed any problems, click on the **Boundary Conditions** toolbar button again.

- Reassign boundary conditions as necessary. The boundary is actually composed of many small boundary segments that default to the condition of **Adiabatic** unless you have imported a glazing system from WINDOW in which case THERM assigns predefined boundary conditions to the glazing. You can change the boundary conditions of each segment of the cross section using the procedure described below in Section 6.2.3.

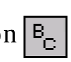
- Define U-factor tags if desired. These tags allow you to specify a group of boundary segments as one unit for which a single U-factor is to be calculated.

When these steps are complete, you have finished assigning the boundary condition definitions and can now perform the heat-transfer calculations for your cross section and then view the results (see Chapter 7, "Calculating Results," for a detailed discussion of the calculation procedure.)

6.2. Assigning Boundary Condition Definitions

The general method for defining boundary conditions is explained below. The example given does not involve any detailed or comprehensive modeling conditions for frames or glazing cavities, which are described in Section 6.5, "Special Boundary Condition Cases."

6.2.1. Generate Boundary Conditions

1. Make sure you save your file, then click on the **Boundary Conditions** toolbar button 

or use the **Draw/Boundary Conditions** menu choice.

2. The program draws an exterior boundary around all the cross section components, indicated by a heavy bold line, as shown in the following figure. This line is actually composed of many boundary segments, each of which is individually assigned a boundary condition.

3. A default boundary condition of **Adiabatic** is automatically assigned to all segments except those on glazing systems imported from WINDOW, which have predefined boundary conditions. The **Adiabatic** boundary condition is defined by a black color. In the predefined Boundary Condition Library that is included with THERM, boundary conditions for segments exposed to a cold environment are indicated with shades of blue; conditions for segments exposed to a warm environment are indicated by shades of red.

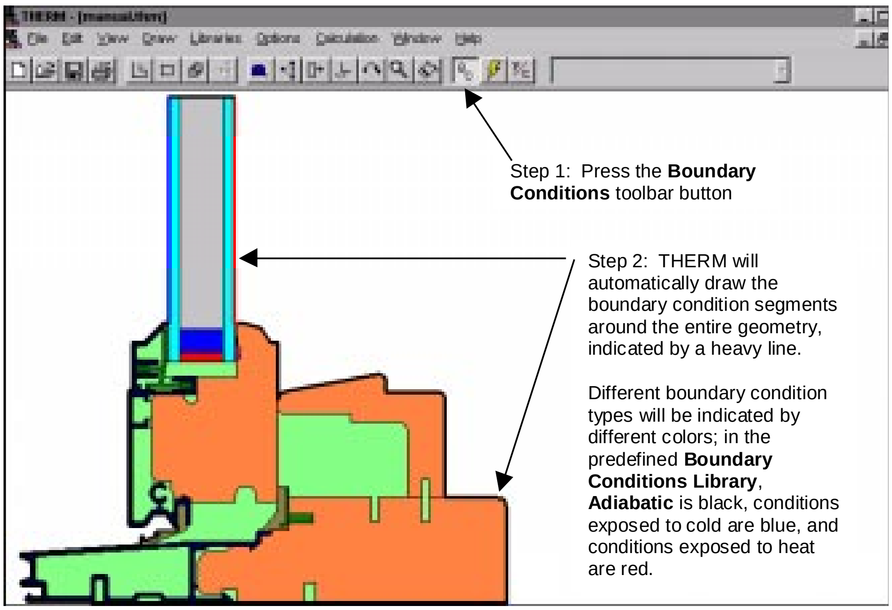


*Figure 6-1. Click on the **Boundary Condition** (BC) toolbar button; THERM will automatically draw the boundary condition segments, indicated by a heavy line.*

6.2.2. Automatic Check of Geometry Definition

When THERM draws boundary conditions, it checks the cross-section geometry to make sure that there are no problems that will cause the thermal calculation to fail. If the program finds such problems, it displays messages describing the problems; the boundary conditions may not be drawn until you correct the problems. Problems that cause the program to issue boundary condition error messages are:

- Noncontiguous polygons, i.e., polygons that are not touching any other polygon in the cross section. The error message will say, "There are materials that are outside of the Boundary Conditions."

- Regions inside the cross section that are not defined as a polygon are classified as voids. Polygons whose edges overlap an adjacent polygon are classified as overlapping. The error message will say, "The geometry contains voids or overlapping regions. The edges surrounding these regions will be highlighted in red. You must fix this problem before simulating." (THERM usually detects this problem as you are drawing, but it does not catch every instance.)

If you get these error messages when you push the **Boundary Conditions** toolbar button, you will have to correct the problems before simulating. See Section 6.3, "Error Detection in THERM" later in this chapter for more information about correcting these problems.

6.2.3. Reassign Boundary Conditions

THERM comes with a predefined **Boundary Conditions Library** that can be used to assign boundary condition to each segment. It is also possible to add new definitions to the **Boundary Conditions Library**, which is discussed in detail in Section 6.4, "Defining New Boundary Conditions". In most cases, you will want to change the boundary condition **Adiabatic** that THERM assigns automatically. THERM cannot perform the simulation without at least two nonadiabatic boundary condition, and will issue a message if this condition is not met. To reassign boundary conditions:

1. Select one or more boundary segments.

   To select ***one*** segment for assigning boundary conditions, place your mouse cursor over the segment and click the left mouse button once. You will see that the segment you have selected is indicated by rounded endpoints.

   To select ***multiple*** boundary segments:

   - For *contiguous* segments, select the first segment (by placing your cursor on the segment and clicking the left mouse button once), hold the **Shift** key down, move your cursor ***in a counter-clockwise direction*** to the last segment, and, keeping the **Shift** key down, click the left mouse button on the last segment and then press **Enter** (or, keeping the **Shift** key down, double click the left mouse button on the last segment) to select the series of segments and display the **Boundary Condition Type** dialog box.

     OR

   - For *noncontiguous* boundary segments, hold the **Ctrl** key down and click on multiple segments, then press **Enter** on the last segment (or, hold the **Ctrl** key down and double-click the left mouse button on the last segment) in order to access the **Boundary Condition Type** dialog box.

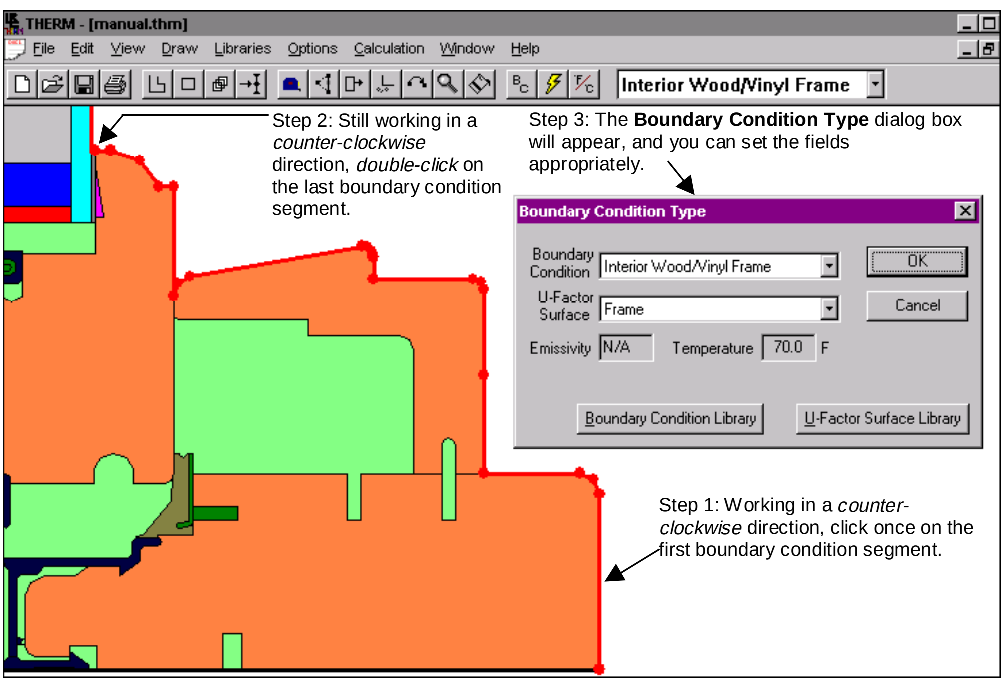


Figure 6-2. Select multiple boundary segments by pressing down the shift key and clicking the mouse on segments as you move in a counter-clockwise direction, OR click on the first segment and then double-click on the last segment, in a counter-clockwise direction; all segments in between will be selected.

2. When you have the selected boundary segments, you can use two methods for reassigning boundary conditions to them:

   - Select the boundary condition definition from the pull-down list on the right-hand side of the toolbar; this list shows the entries in the **Boundary Condition Library**.

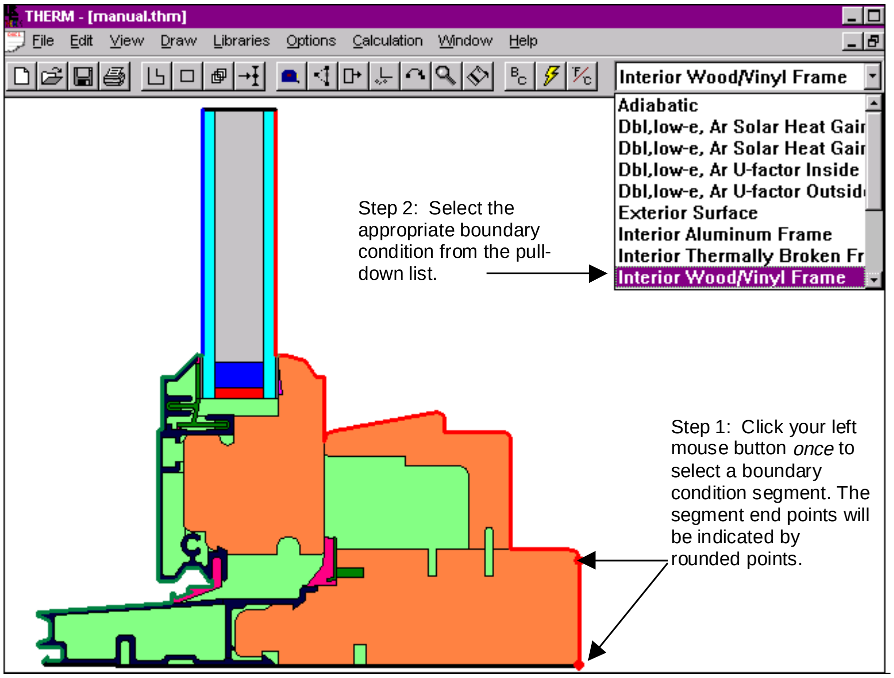


Figure 6-3. One method for defining the boundary condition properties is to click on one or more boundary segments and pick the desired boundary condition definition from the pull-down list.

   OR

   - Press **Enter** after you have selected the boundary condition segment(s), which accesses the **Boundary Condition Type** dialog box. This method has the advantage of allowing you to change the **U-factor Surface** tags (see Section 6.2.4, "Define U-factor Surface Tags") and also accessing the **Boundary Condition Library** in addition to selecting the boundary condition. (Alternatively, you can select the boundary segment with one left mouse click; and then select the **Library/Set Boundary Condition** menu choice to access the **Boundary Condition Type** dialog box, or select the **Library/Boundary Condition Library** menu choice to access the **Boundary Conditions** library dialog box.)

   Select the appropriate values from the pull-down lists for **Boundary Condition** and **U-Factor Surface**. The choices in the **Boundary Condition** list show all the entries in the **Boundary Condition Library**, and the choices in the **U-Factor Surface** list show all the entries in the **U-Factor Surface Library**. These libraries can be accessed directly from this dialog box using the two buttons at the bottom of the box. If you have imported a glazing system, the boundary conditions for that glazing system will also appear in the **Boundary Condition** list (the glazing system name appears as a prefix to the title) for both **U-factor** and **Solar** conditions.

   You can access the **Boundary Condition Library** (shown in the following figure) to view the properties associated with a boundary condition by clicking the **Boundary Condition Library** button at the bottom of the **Boundary Condition Type** dialog box. The predefined boundary conditions that are provided with THERM cannot be edited, but you can add new boundary conditions to the library (see Section 6.4, "Defining New Boundary Conditions" later in this chapter).

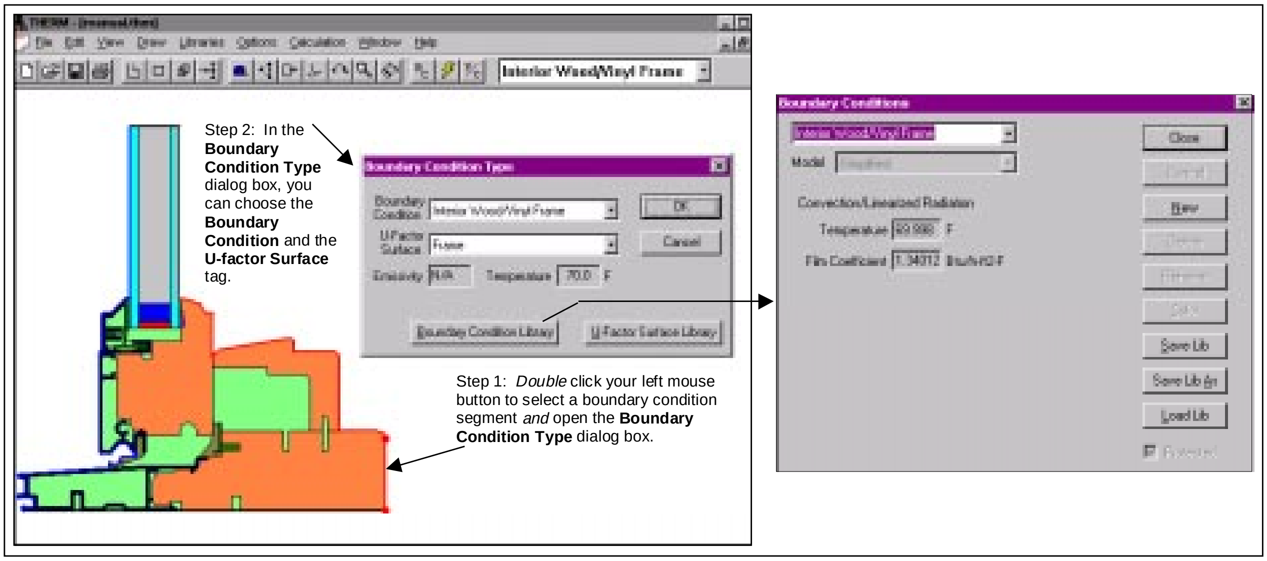


Figure 6-4. Another method for assigning boundary conditions is to double-click on a segment to access the ***Boundary Condition Type*** *dialog box where you can select the* ***Boundary Condition*** *as well as the* ***U-factor Surface*** *tag name.*

Glazing systems that are imported from WINDOW have predefined boundary conditions; THERM assumes that the outside surface is on the left unless you flip the glazing system horizontally after it is imported. Four boundary conditions are imported with the glazing system:

- *U-factor Inside Film, U-factor Outside Film:* calculated under environmental conditions equivalent to ASHRAE winter.
- *Solar Heat Gain Inside Film, Solar Heat Gain Outside Film:* calculated under environmental conditions equivalent to ASHRAE summer.

These boundary conditions show the WINDOW glazing system name as a prefix. For example, the name of an inside glazing system boundary condition would be "Dbl, low-e, Ar U-factor Inside Film" if the name of the glazing system imported from WINDOW was "Dbl, low-e, Ar."

3. Continue to define all the boundary conditions. When you are finished your cross section should look similar to the following figure. You can see boundary conditions more clearly if you turn off the material colors (go to the **View** menu and uncheck the **Material Colors** choice; then, you will see only the polygon outlines, and the boundary conditions appear prominent). Similarly, you can turn off the display of the boundary conditions by unchecking the **Boundary Condition** choice in the **View** menu.

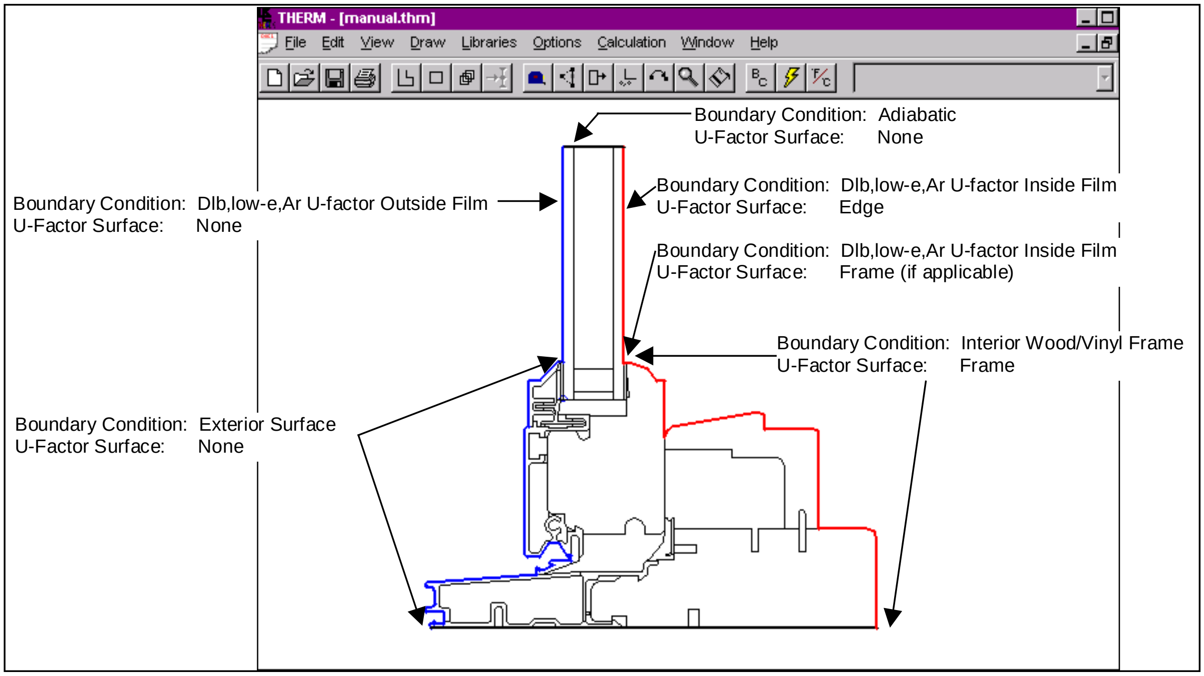


Figure 6-5. Final boundary condition configuration for a simplified model

6.2.4. Define U-factor Surface Tags

U-factor surface tags are used to label and group boundary segments for the program's U-factor calculation. Without these tags the program will not calculate U-factors. The U-factor is a measure of the heat transfer through a cross section under specific environmental conditions. The U-factor calculation integrates the heat flux for the boundary segment (or group of segments that have been given the same tag), divides that flux by the projected length of the segment(s) and the defined temperature difference, and displays a U-factor. A single U-factor tag can be assigned to a group containing as many segments as you wish. You can create as many tags or groups as you wish. A U-factor will be calculated for each unique tag name. U-factors can be calculated for a group of boundary segments even if individual segments within the group were defined with different boundary conditions.

It is common practice in the analysis of building components to define the U-factor on interior (room) boundary surfaces. Interior and exterior surfaces are often separated by adiabatic boundaries. The U-factor of adiabatic boundaries (those having no heat flow) will be zero, so there is no need to tag these surfaces. If U-factors are desired for the exterior boundary surfaces, these surfaces should be tagged separately from interior surfaces. THERM assumes that no heat is stored in the cross section, so all energy that enters the cross section on the interior surface leaves through the exterior surface. If the full length of the interior and exterior surfaces are given the same **U-factor Tag**, the resulting U-factor will be zero.

The choices for the **U-factor Tag** pull-down list are the values that come with THERM:

- *None:* Use **None** for adiabatic boundary segments or any segment where a U-factor result is not desired.
- *Frame:* Use **Frame** for interior frame boundary segments and for glazing boundary segments that are between the frame and the sight line.
- *Edge:* Use **Edge** for all the glazing system boundary segments that are above the sight line.

You can make new values in the **U-factor Names Library** by clicking on the **U-factor Surface Library** button on the **Boundary Condition Type** dialog box. Click on **Add** and type in a unique **U-factor Surface Tag** name, and the program will add that name to the **U-factor Names Library**.

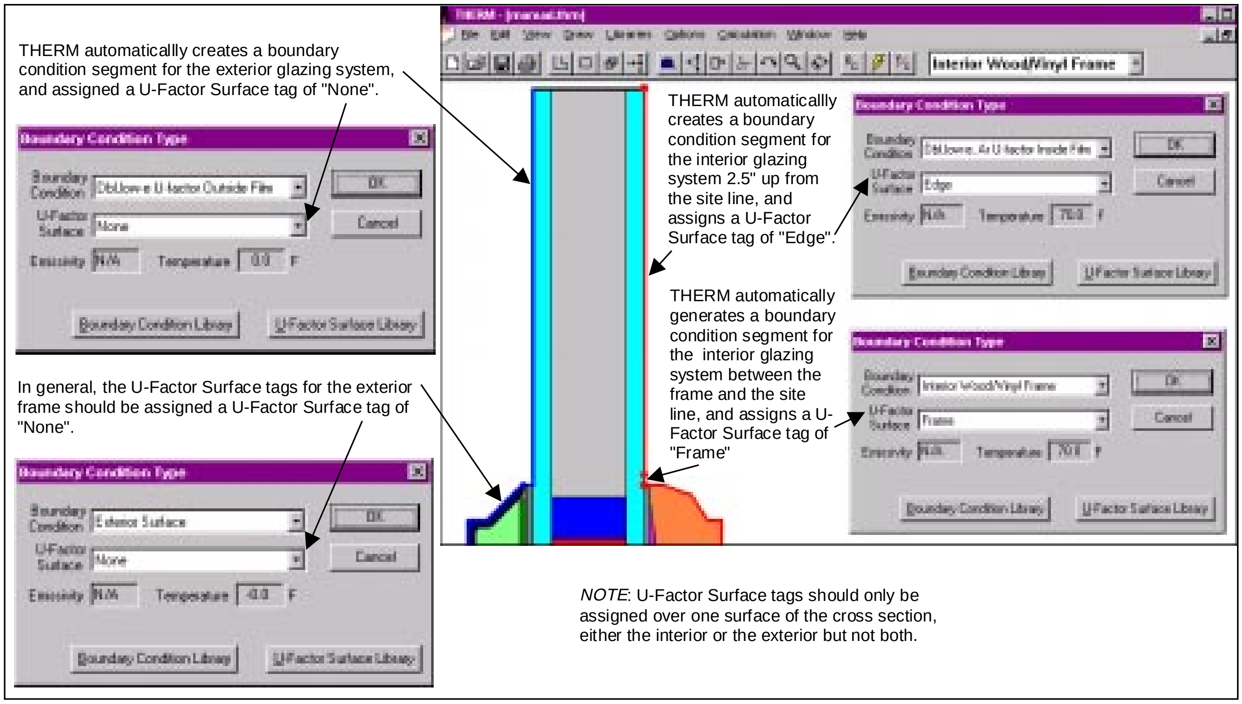


Figure 6-6. Assigning U-Factor Surface tags to different boundary segments. THERM automatically makes two boundary segments on the interior surface of the glazing system, one from the frame to the sight line (with a U-Factor Surface tag of "Frame") and one from the sight line up to 2.5" of the glass (with a U-Factor Surface tag of "Edge").

6.3. Error Detection in THERM

Defining the boundary conditions invokes several features in THERM which detect errors made during the drawing process. The error detection features are intended to help you fix questionable (or “bad") points and find small voids and overlaps in the geometry so that the automatic mesh generator can create a well formed mesh. Other error detection features in THERM that identify invalid polygons and polygons which cause the automatic mesh generator to fail are discussed in Section 7.3.

THERM's error-detection algorithms rely on the definition of two minimum allowable distances between two adjacent points in a cross section:

- **Floating point tolerance:** The distance between two points at which THERM will automatically merge the points, set to 0.01 mm.
- **Checking tolerance:** The distance between two points at which THERM will mark the points as bad, set to 0.1 mm.

When checking for geometry errors, THERM does the following:

- First, THERM checks the polygon edges because the mesher needs exactly matching points on any polygons that are adjacent to each other. If a point exists on the edge of a polygon but not on the edge of an adjacent polygon, and if the adjacent edge is less than the floating point tolerance (0.01 mm), THERM automatically adds a point to the adjacent edge and merges the points so that there is an identically placed point on both polygons. If the distance between the adjacent edges is greater than the floating point tolerance, the program does nothing, and an unmatched edge is created resulting in either a void or an overlapping region, as shown in Figure 6.7.
- Next, THERM checks to see if any points are closer together than the checking tolerance (0.1 mm), and marks such points with red circles, indicating they are "bad" points. It is important for you to check and correct these marked areas before the simulation continues.

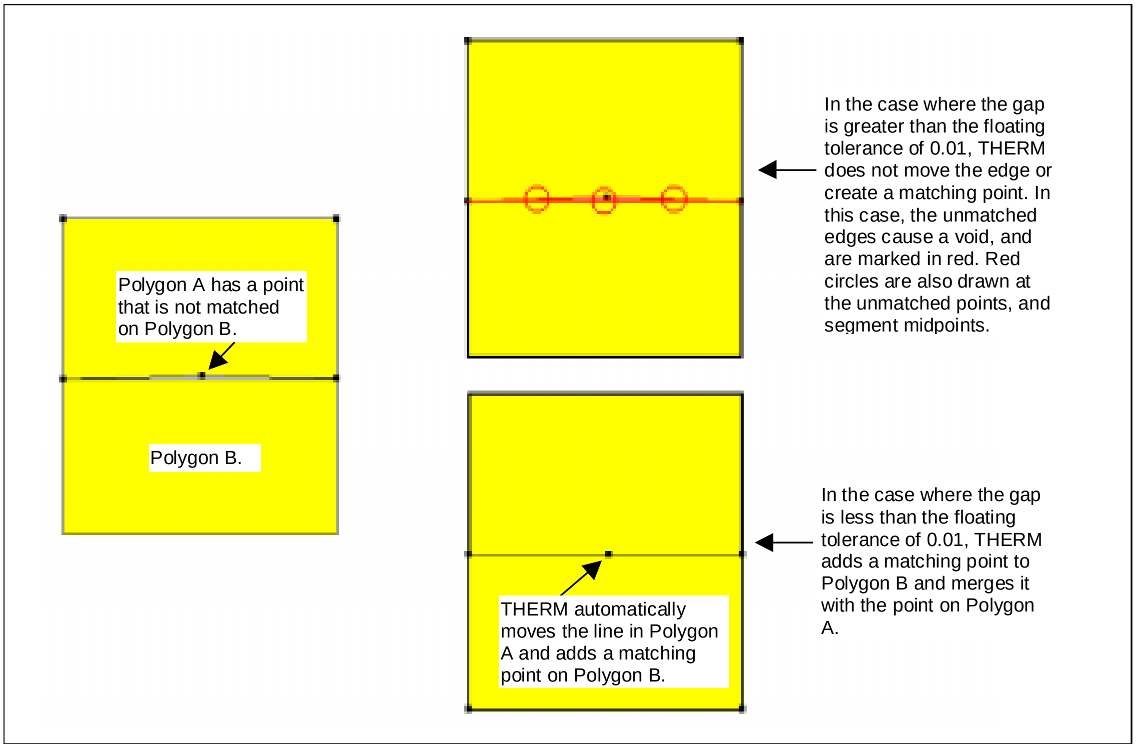


Figure 6-7. The size of the void in a drawing will determine how THERM will handle the error checking.

6.3.1. Finding and Fixing Bad Points

As discussed above, THERM marks points as bad if they are separated by a distance of less than the checking tolerance (0.1 mm). If these points are mistakes or flaws in the model they should be fixed but in some cases they may be intentional details that should be left alone. Because you may want to keep these points in the model, THERM draws red circles around the points and displays a message, shown in Figure 6-8, which gives you the option of having the program either mark the points so that you can evaluate them or automatically adjust them:

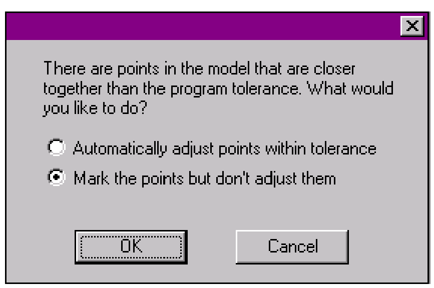


Figure 6-8. This message may appear when you create the boundary condition

- *Mark the points but don't adjust them:* This is the default option and it is recommended that you choose this option for your first attempt at assigning the boundary conditions. It allows you to check the circled points and either correct the points manually or leave them in the model if you want to keep them.

  Occasionally points exist on the edges of both adjacent polygons, but they are not exactly aligned. In this case the program adds matching points to the edges of each polygon resulting in points on both polygons that are very close together (which may be detected as bad points). In this case, you need to delete one of the points, remembering to delete the point on both polygons. This can be done by selecting one polygon, holding down the shift key, selecting the other polygon and then using the **Draw/Edit Points** menu choice to delete the point from both polygons in one step. If you do not delete the point on both polygons THERM will automatically recreate it when you reassign the boundary conditions. Be careful in choosing which point to delete that you select the one that will allow the geometry of the model to be accurately represented.

If you choose to have the program mark the bad points, you may want to improve the clarity of the display by turning off the material colors. Go to the **View** menu and uncheck the **Material Colors** choice. Then you will see only the outlines of the polygons, and the circled points will become very obvious. You can zoom in (click the right mouse button) on the circled area to see if you need to fix the points.

Sometimes there will be a red circle in an area that looks fine. If you select the vertices attached to that point one at a time and move them a short distance (less than the sticky distance) from the point and release the point, the vertices will snap to a common point and the problem point will be removed. If your model only contains areas like this you can use the Automatically Adjust Points within Tolerance options without negative consequences.

If there are fine details containing points less than 0.1 mm apart that you want to preserve in your model, you can turn off the display of the red circles using the **View/Bad Point** command. However, even though you can't see the circles they are still there. If you want to remove the red circles, they can be cleared using the **Draw/Clear Bad Points** command.

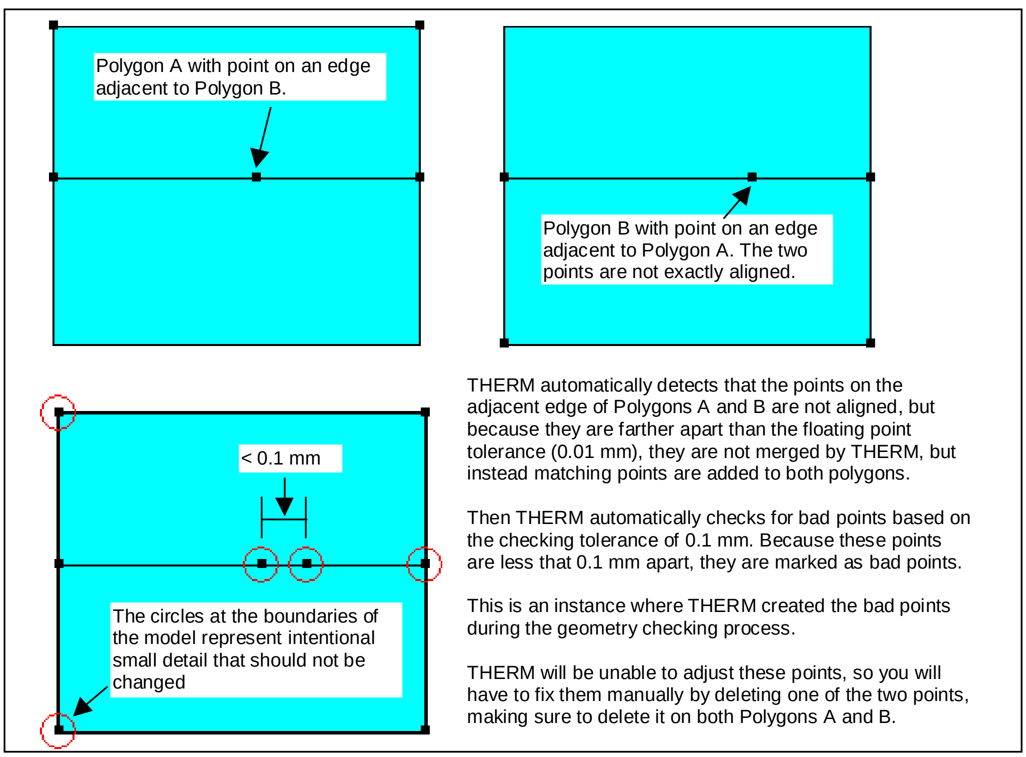


Figure 6-9. An instance where THERM created bad points in the process of checking for matching points on adjacent polygons.

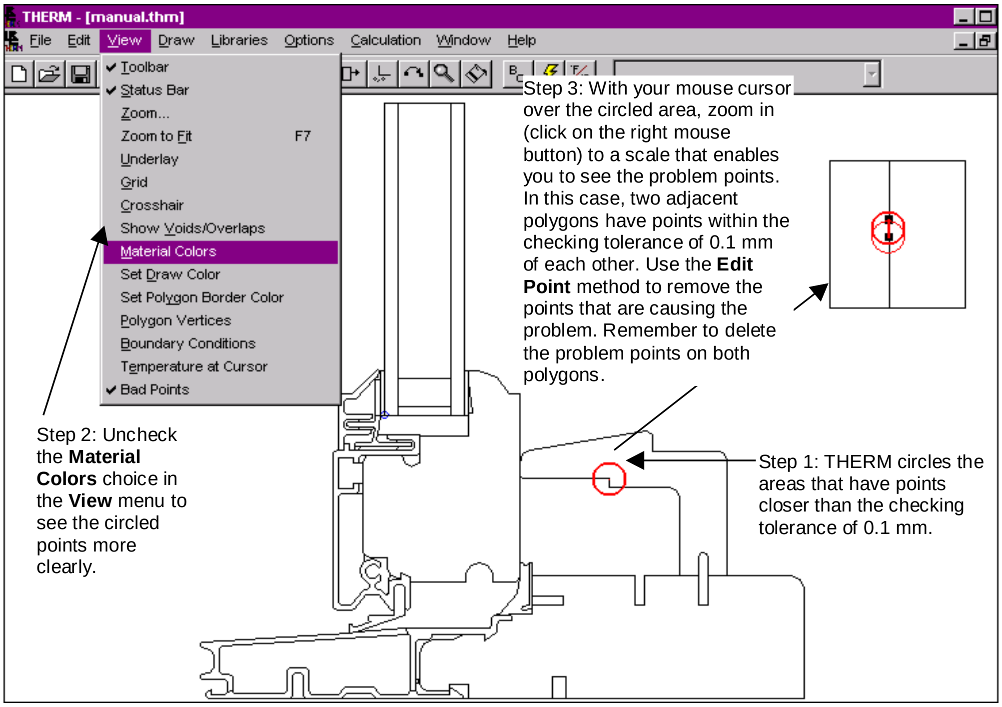


*Figure 6-10. To view the circled "bad" points more clearly, "uncheck" the **Material Colors** choice from the **View** menu.*

- *Automatically adjust points within tolerance:* When this option is chosen, THERM will automatically merge any two points that are within the checking tolerance of 0.1 mm. In cross sections with fine detail the automatic fix feature can make unwanted changes in your model. It is strongly recommended that you save your work before using this feature. Once this option is selected it cannot be undone with the undo command.

  Sometimes when using the automatic adjust option you will get a message saying that THERM cannot fix all the bad points, as shown in Figure 6-11. The unfixed points will be marked and you must fix them before the simulation continues.

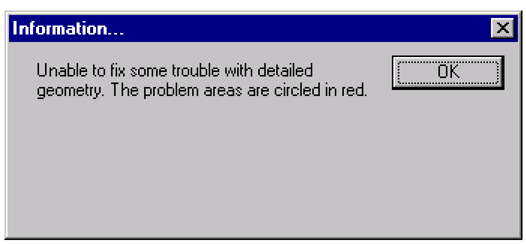


Figure 6-11. This message may appear if you select the automatic adjustment option and THERM cannot correct the problem.

The existence of bad points does not keep THERM from assigning the initial boundary conditions. When you are adjusting the model to correct bad points, if you do not make changes to the segments that make up the boundary, the boundary conditions will not be deleted and you can continue with the simulation process without reassigning the boundary. This is the recommended way to proceed as it ensures that you have correctly fixed all problems in the model. It is recommended, however, that you do recreate the boundary conditions to ensure that you have correctly fixed all problems in the model. If you do recreate the boundary conditions make sure to delete the existing red lines and circles using the **Draw/Clear Bad Points** menu option. THERM will remember the boundary condition assignments so you will not need to redefine the boundary segments.

6.3.2. Finding Voids and Overlaps

Defining the boundary conditions also causes THERM to check the model for small voids and overlaps that are caused by unmatched polygon edges as discussed in Section 6.3, "Error Detection in THERM." (Large voids and overlaps can be detected using the **View/Voids and Overlaps** featured discussed in Chapter 5). An unmatched edge is an edge on a polygon that does not have a matching edge on an adjacent polygon, which indicates that there is either a void or an overlap in that area. THERM marks small voids and overlaps by drawing the unmatched edge in red with a circle at its midpoint and displaying the warning message as shown in Figure 6-12. These areas must be fixed before the simulation process can continue.

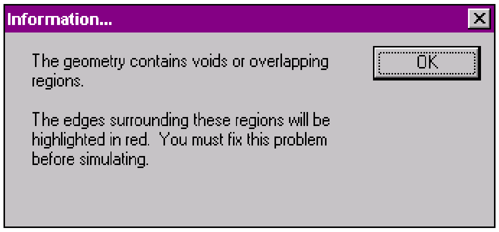


Figure 6-12. THERM issues a warning if it detects voids or overlapping regions when trying to define boundary conditions.

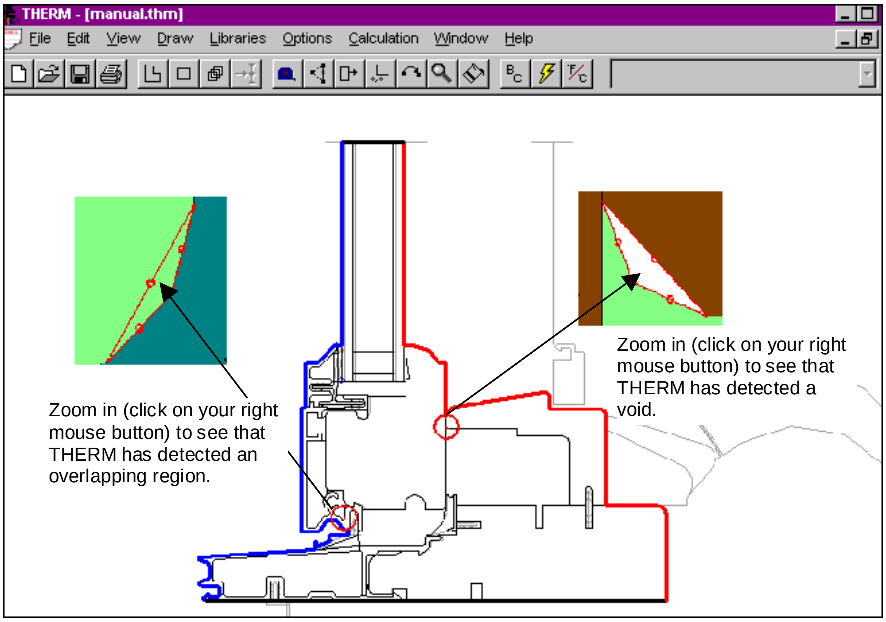


Figure 6-13. Voids and overlaps are indicated by circles once the boundary conditions have been assigned.

6.4. Defining New Boundary Conditions

THERM includes a Boundary Conditions Library with a set of predefined boundary conditions that are protected from editing (the input values are disabled). If you do not find the boundary condition you need in the **Boundary Condition** pull-down list, you can add it to the **Boundary Condition Library** using the following technique.

1. Open the **Boundary Condition Library** by selecting the **Library/Boundary Condition Library** menu choice; the **Boundary Conditions** dialog box will be displayed. The predefined boundary conditions cannot be edited, but you can use the pull-down list at the top to view the characteristics of any of the entries.

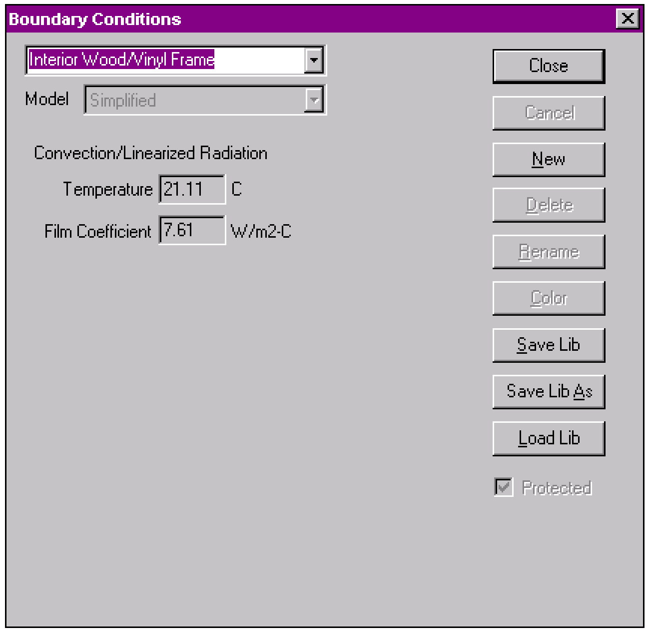


Figure 6-14. The default Boundary Condition Library that comes with THERM has boundary condition definitions that cannot be edited.

2. To make a new boundary condition, click on the **New** button.

3. You will be asked for the name of the new boundary condition. Type a name that is not already in the library and click on the **OK** button.

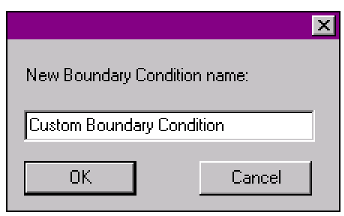


Figure 6-15. Type a unique name for the new boundary condition.

4. Click on the **Color** button to change the color from the default value of blue.

5. Now you are presented with the same **Boundary Conditions** dialog box, but all the fields are available for you to edit, so you can create your custom boundary condition. Different fields are available for editing depending on the **Boundary Condition Model** that is chosen, i.e., simplified, comprehensive, or radiation enclosure surface. These different types are discussed in detail in the following sections. Fill out the input values as appropriate.

6. When you have defined the characteristics of the boundary condition, click on the **Save Lib** button; the boundary condition will be saved permanently to the default library (named **bc.lib**). If you forget to save, THERM will ask if you want to save when you exit the program (unless you turn this feature off using **Options/Preferences/Prompt for saving libraries on program exit**).

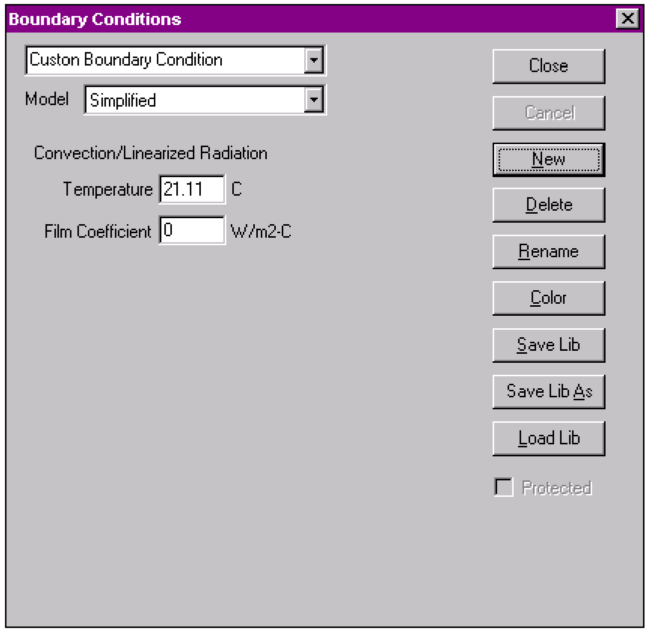


*Figure 6-16. To define a new boundary condition, click on the **New** button, and the **Boundary Conditions** dialog box will appear, allowing you to define the characteristics of the new boundary condition.*

In the **Boundary Conditions** dialog box, you can set the **Model** pull-down list to **Comprehensive**, and the input values that will be available are the following:

Convection — Define the convection values for:

- *Temperature* -- surrounding temperatures for convective heat transfer
  Units: °F (IP); °C (SI)

- *Film coefficient* -- film coefficients to use for convective heat transfer
  Units: Btu / h-ft^2-°F (IP); W / m^2-°C (SI)

Constant Heat Flux — Used to model a constant heat flux on the surface, such as solar gain

- *Flux* -- constant flux value
  Units: Btu / h-ft^2 (IP); W / m^2 (SI)

Radiation — Used to define the radiation component of the model

- *Enclosure Model:* use this if you are defining boundary conditions for the cross-section elements that "see" the external radiation enclosure.

- *Black-Body Radiation:* use this if you want to model radiation exchange with a black body as an alternative to drawing a radiation enclosure. For this case you have to define the values for view factor and temperature (Ti).

Constant Temperature — Used to define the temperature at the surface being specified

- *Temperature*: Units: °F (IP); °C (SI)

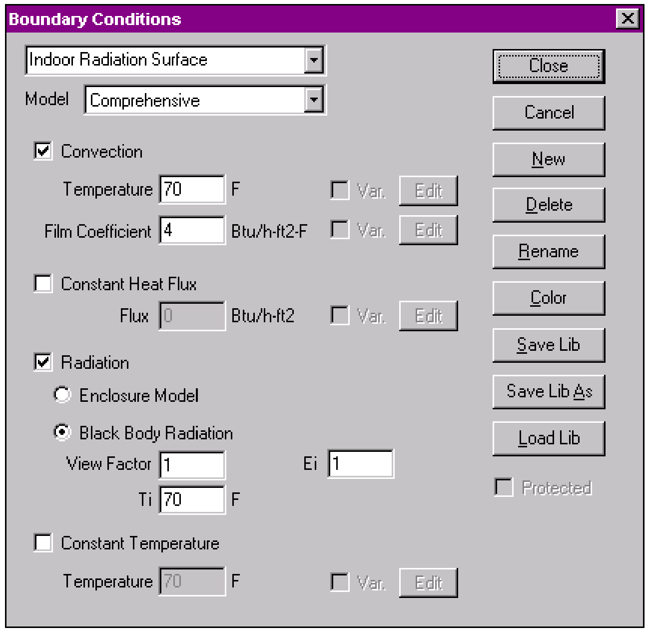


Figure 6-17. Sample boundary condition library entry for a comprehensive model

6.5. Special Boundary Condition Cases

There are two special boundary condition cases in THERM, for the following conditions:

- Detailed frame cavity radiation model
- External Radiation Enclosure

6.5.1. Frame Cavities with Detailed Radiation Model

If you used a detailed radiation model within a frame cavity, then, when you create the boundary conditions for that cavity, THERM will automatically highlight the cavity surfaces in red and label them **Frame Cavity Surface** (shown in the following figure).

Sometimes frame cavities are broken up into multiple cavities to better model convection heat transfer. If this is done with a detailed frame cavity model THERM treats the sub-cavities individually for convection purposes but merges them for radiation purposes. The red boundary will show this.

If you change the cavity to another type of material after the boundary conditions are drawn, you must redefine the boundary conditions.

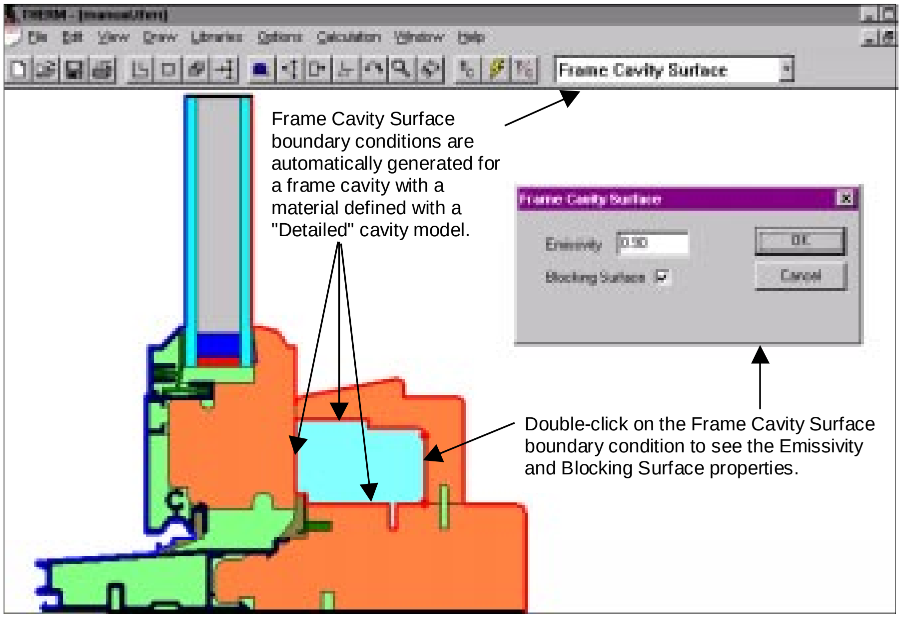


Figure 6-18. When the frame cavity is assigned a material with a Detailed cavity model, THERM automatically assigns boundary conditions to the cavity.

To see the **Frame Cavity Surface** properties, double-click on one of the red segments; a dialog box will appear with the following values:

Emissivity This value is the emissivity of the frame cavity surface, automatically determined by THERM based on the emissivity of the surrounding materials as defined in the **Material Library**. You can override the definition by changing the value in this input box. If the emissivity value is changed in the **Material Library** after the boundary conditions are drawn, the change will not take effect until the boundary conditions are redrawn (by clicking on the **Boundary Conditions** toolbar button or pressing **F10**.)

Blocking Surface This check box indicates whether the surface that you are defining should be checked automatically by the program to determine if it blocks radiation. A radiation blocking surface is a surface that interferes with the radiation that would otherwise travel between surfaces, as shown in the following figure. Checking this box does not automatically make it a blocking surface, it just tells THERM to determine whether or not the surface is blocking. If the box is not checked, the program will not make this determination. **Default:** Checked, which means that all surfaces will be checked by THERM to see if they are blocking. This default is recommended because it guarantees that the radiation is modeled correctly. However, for simple rectangular surfaces, there is no need to check for blocking surfaces.

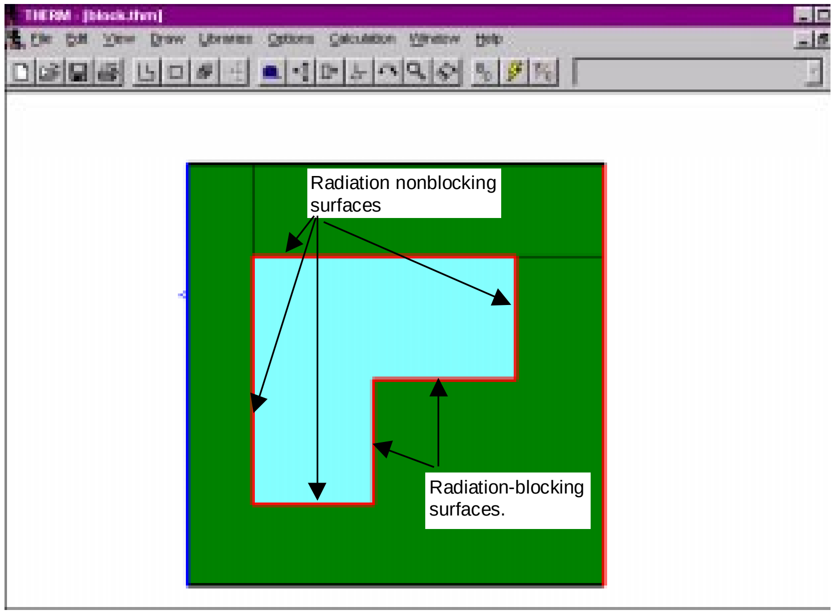


Figure 6-19. An example of radiation-blocking surfaces

6.5.2. External Radiation Enclosures

If you assign a radiation enclosure material to a polygon, then, when you create the polygon's boundary conditions, THERM will automatically assign boundary conditions named **Radiation Surface** (shown in the following figure).

To see the **Radiation Surface** properties, double-click on one of the boundary conditions; a dialog box will appear showing the emissivity and temperature, as well as a checkbox indicating whether the surface is blocking radiation.

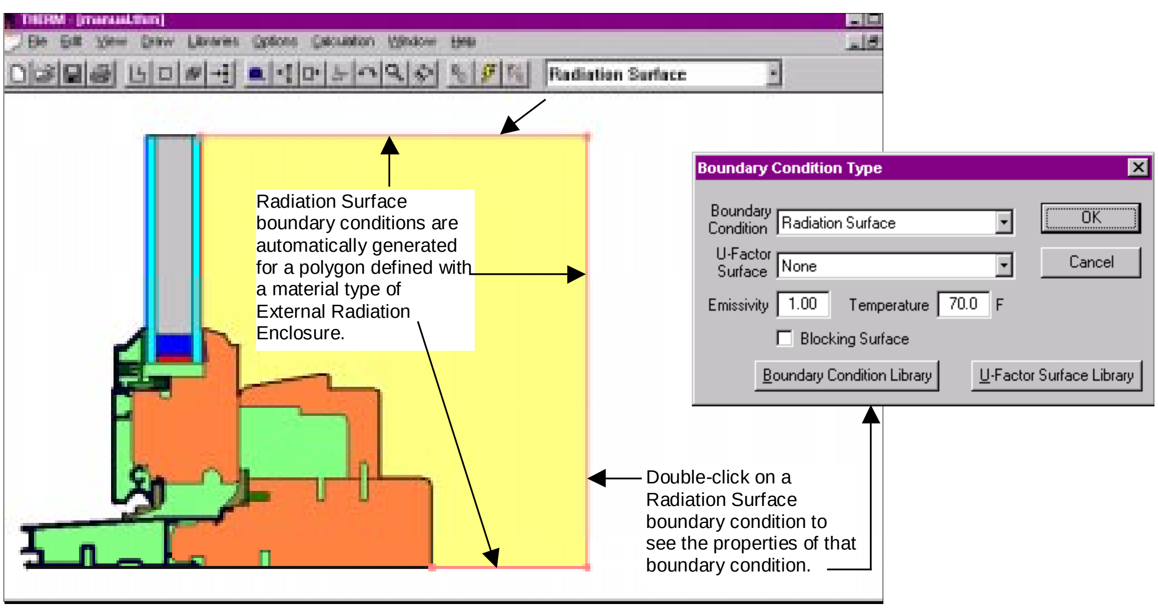


Figure 6-20. THERM automatically assigns Radiation Surface boundary conditions to the external radiation enclosure surfaces.

In order for the program to model an external radiation enclosure, it may be necessary to change the boundary conditions of the cross section elements that "see" the enclosure, in this case the interior frame and glazing assembly elements.

Changing these boundary conditions requires you to first create custom **Boundary Condition Library** records (see Section 6.4, "Defining New Boundary Conditions") that have the following definitions, as shown in Figure 6-21:

Model	Set to **Comprehensive**.
Convection	Define the appropriate temperatures and film coefficients for each element type. Make sure that the film coefficient only accounts for convection and does not contain a radiation component. (The film coefficients in the simplified boundary condition are a combination of convective and radiative effects.)
Radiation	Set to **Enclosure Model**.

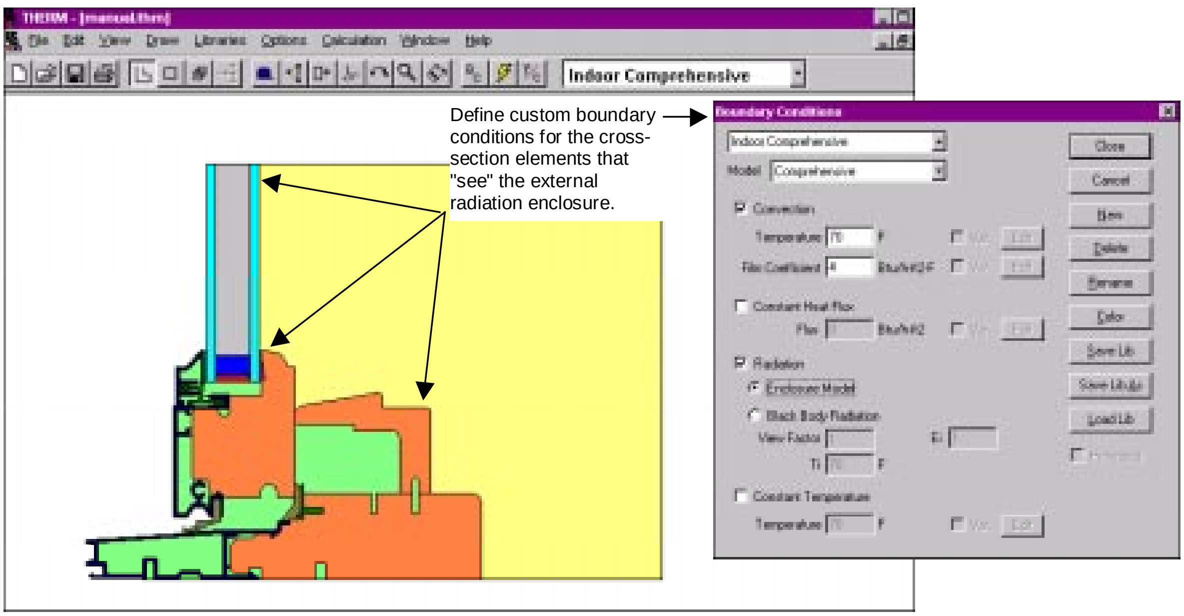


*Figure 6-21. Define new boundary conditions for the elements that "see" the radiation enclosure with the **Comprehensive** model.*

When you have defined the new boundary condition in the library, you need to assign it to the appropriate boundaries, as shown in Figure 6-22.

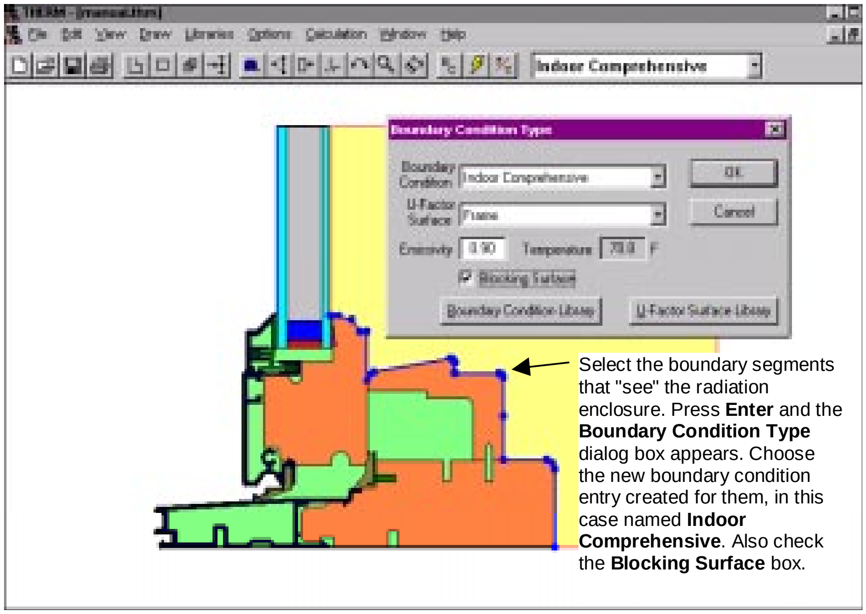


Figure 6-22. Set the boundary conditions for the surfaces that "see" the radiation enclosure.